



On the application of the finite element method in dynamic and static problems of the mechanics of a deformable body

Buronov Sunatullo Aslonovich¹, Avezov Alijon Xayrullyevich², Umirzoqov Jurabek Umrzoqovich³

¹Assistant of the Department "Mechanics". Bukhara Engineering and Technology Institute, Uzbekistan

²Senior Lecturer of the Department "Mathematics". Bukhara State University. Uzbekistan

³Student group 8-15TJMS. Bukhara Engineering and Technology Institute, Uzbekistan

Article History

Received: 17 April 2019

Accepted: 09 June 2019

Published: July 2019

Citation

Buronov Sunatullo Aslonovich, Avezov Alijon Xayrullyevich, Umirzoqov Jurabek Umrzoqovich. On the application of the finite element method in dynamic and static problems of the mechanics of a deformable body. *Discovery*, 2019, 55(283), 322-327

Publication License



© The Author(s) 2019. Open Access. This article is licensed under a [Creative Commons Attribution License 4.0 \(CC BY 4.0\)](https://creativecommons.org/licenses/by/4.0/).

General Note



Article is recommended to print as color digital version in recycled paper.

ABSTRACT

The paper deals with the solution of static and dynamic problems of the mechanics of a deformable solid (MDTT) by the finite element method. Various approaches to solving the MDTT problem for modern finite element methods (FEM) are given.

Keywords: solid mechanics, rigidity, stability, matrix formulation, stiffness matrix, discretization

1. INTRODUCTION

Finite element method

The main method of modern structural mechanics and mechanics of a deformable solid body that underlies the vast majority of modern software systems designed to perform calculations of deformable bodies on a computer.

Mechanics of a deformable solid body

Set of sciences of strength, stiffness and stability of deformable bodies. But the range of its application is extremely wide: construction and engineering, hydro and aerodynamics, mining and the latest technology, as well as various problems of mathematical physics - heat conduction, filtration, wave propagation, etc.

The finite element method was first applied in engineering practice in the early 50s. XX century. At an early stage, the FEM formulations were based on the principles of structural mechanics, which limited its scope. And only when the foundations of the method were formulated in a variation form, it became possible to extend it to many other tasks. The rapid development of FEM proceeded in parallel with the progress of modern computer technology and its application in various fields of science and engineering practice [1-6].

A significant contribution to the development of FEM was made by Ioannis Argyris. For the first time, he gave a general matrix formulation of the calculation of rod systems based on fundamental energy principles, defined a compliance matrix, and also introduced the notion of a stiffness matrix (as an inverse compliance matrix). Argyris is one of the founders of the finite element method. In 1956, his theoretical developments were used in the construction of the Boeing 747. The works of Argyris and his staff, published in the period 1954–1960, gave a starting point for the matrix formulation of well-known numerical methods and the use of computers in structural calculations.

The first work, in which the modern concept of FEM was presented, refers to 1956. American scientists M. Turner, R. Cliff, G. Martin and L. Topp, solving the flat problem of the theory of elasticity, introduced an element of a triangular form, for which a stiffness matrix was formed and the nodal force vector. The name - *the finite element method* introduced in 1960 by R. Klaff.

The appearance of the mathematical theory of finite elements belongs to the seventies. A significant contribution to the development of the theoretical foundations of FEM was made by scientists of the CIS [7, 8].

The period of the last decades is especially characteristic of the development and application of FEM in such areas of continuum mechanics as optimal design, taking into account nonlinear behavior, structural dynamics, etc.

Basic concepts by finite element method

In real structures, there are almost always complex shapes, which also consist of different materials. The finite element method is the most popular numerical method for solving problems of designing structures of complex shapes.

Sampling Analysis by the finite element method begins with discretization of the studied area (the problem area) and dividing it into grid cells. Such cells are called finite elements (Fig. 1 and Fig. 2).

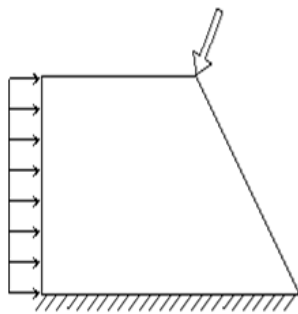


Figure1 Settlement scheme

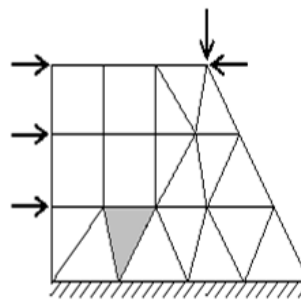


Figure 2 Object discretization

Finite elements may have a different shape. In contrast to a real structure in a discrete model, finite elements communicate with each other only at separate points (nodes) by a certain finite number of nodal parameters.

The selection of suitable elements with the required number of nodes from the library of available elements is one of the most important decisions that the user of a package of course elemental analysis has to make. The designer also has to set the total number of elements (in other words, their size).

The main problem of FEM is grid construction, especially for an object with complex geometry. Creating three-dimensional finite element meshes is usually a laborious and painstaking process.

The classic form of the finite element method is called the h-version. As a function of the form in this method, piecewise polynomials of fixed degrees are used, and an increase in accuracy is achieved by reducing the size of the cell. In the p-version, a fixed grid is used, and accuracy is increased by increasing the degree of the form function. The general rule is that the greater the

number of nodes and elements (in the h-version) or the higher the degree of the form function (p-version), the more accurate the solution is, but the more expensive it is from a computational point of view. One of the CAD systems in which the FEM p-version is implemented is Pro / Engineer (CREO).

Ensemble

An ensemble or assembly is the union of individual elements in a finite element mesh. From a mathematical point of view, enfeblement consists in combining the stiffness matrices of individual elements into one global stiffness matrix of the whole structure. In this case, two systems of element numbering are used essentially: local and global. Local numbering is a fixed numbering of nodes for each type of finite elements in accordance with the entered local coordinate system on the element. The global numbering of the nodes of the whole structure can be completely arbitrary, as well as the global numbering of finite elements. However, there is a one-to-one correspondence between local numbers and global node numbers, on the basis of which the global system of finite element equations is formed [11].

2. APPROXIMATION

FEM refers to discrete analysis methods. However, unlike numerical methods based on the mathematical discretization of differential equations, the FEM is based on the physical discretization of the object under consideration. A real construction as a continuous medium with an infinitely many number of degrees of freedom is replaced by a discrete model of interconnected elements with a finite number of degrees of freedom. Since the number of possible discrete models for the continual region is infinitely large, the main task is to choose a model that best approximates this region. The essence of the continuum approximation by FEM is as follows:

- The area under consideration is divided into a certain number of FE; the family of elements throughout the area is called a system or a grid of finite elements;
- It is assumed that QEs are interconnected at a finite number of points - nodes located along the contour of each of the elements;
- For each QE an approximating polynomial is given.

Approximating functions

Approximating polynomial for one-dimensional CE:

$$u(x) = \sum_{i=0}^r \alpha_i x^i$$

An example for one-dimensional and two-dimensional CE is shown in Fig. 3, Fig. 4 and Fig. 5.

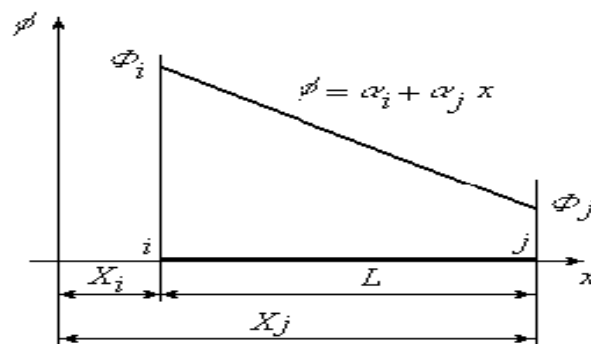


Figure 3 Example for one-dimensional CE

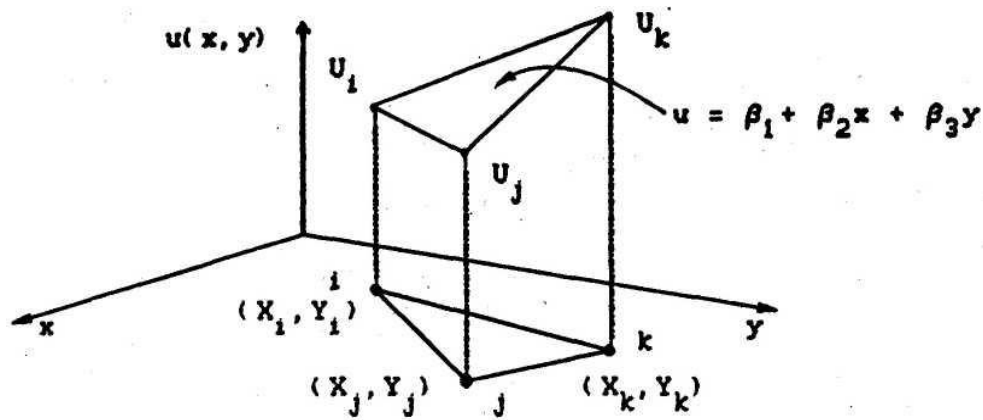


Figure 4 Example for two-dimensional CE

The main types of CE:

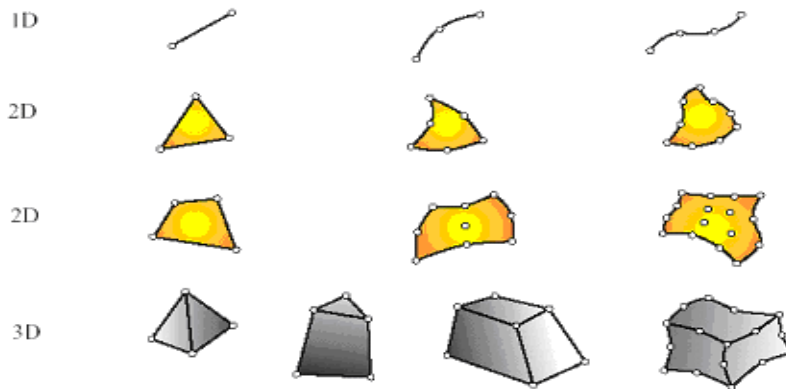


Figure 5 Types of CE

Finite elements can be described by one, two or three spatial coordinates depending on the dimension of the problem for which they are intended. The corresponding number of internal or local coordinates is called the element's own dimension. In dynamic analysis, time is considered as an additional dimension.

Approximate polynomial of the second order

$$u^e(x, y) = \beta_1 + \beta_2 x + \beta_3 y + \beta_4 x^2 + \beta_5 xy + \beta_6 y^2$$

The degree of the approximating polynomial determines the number of nodes that the element must have — it must be equal to the number of unknown coefficients α_i , included in the polynomial. The sought functions within each CE (for example, the distribution of displacements, deformations, stresses, etc.) with the help of approximating functions are expressed in terms of nodal values, which are the main unknown FEMs.

Seeking approximation function:

$$u(\bar{x}) = \sum_{i=0}^r h_i(\bar{x}) q^i$$

$h(x)$ coordinate / basic functions, so-called form function; q - unknown coefficients (values in nodes).

In matrix form: $\bar{U}(\bar{x}) = \bar{H}\bar{U}$

Approximation, as a rule, gives an approximate, and not exact, description of the actual distribution of the unknown quantities in the element. Therefore, the results of design calculations in the general case are also approximate. The question of the accuracy, stability and convergence of the solutions obtained by the FEM can be raised naturally.

By precision is meant the deviation of the approximate solution from the exact or true solution. Stability is primarily determined by the growth of errors in the performance of individual computational operations. An unstable solution is the result of an unsuccessful selection of approximating functions, a "bad" division of the region into KE, incorrect representation of boundary conditions, etc. By convergence, we mean gradual approximation of successive solutions to the limiting one, such as the dimensions of elements, degree of approximating functions, etc. In this sense, the concept of convergence is similar to the value it has in ordinary iterative processes. Thus, in a convergent procedure, the difference between subsequent decisions decreases, tending to zero in the limit.

The concepts listed above are illustrated in fig. 6. Here the abscissa denotes the degree of refinement of the parameters of the discrete model, and the ordinate determines the approximate solution obtained by this refinement. The graph shows a monotonous type of convergence, in which the accuracy of the solution increases smoothly.

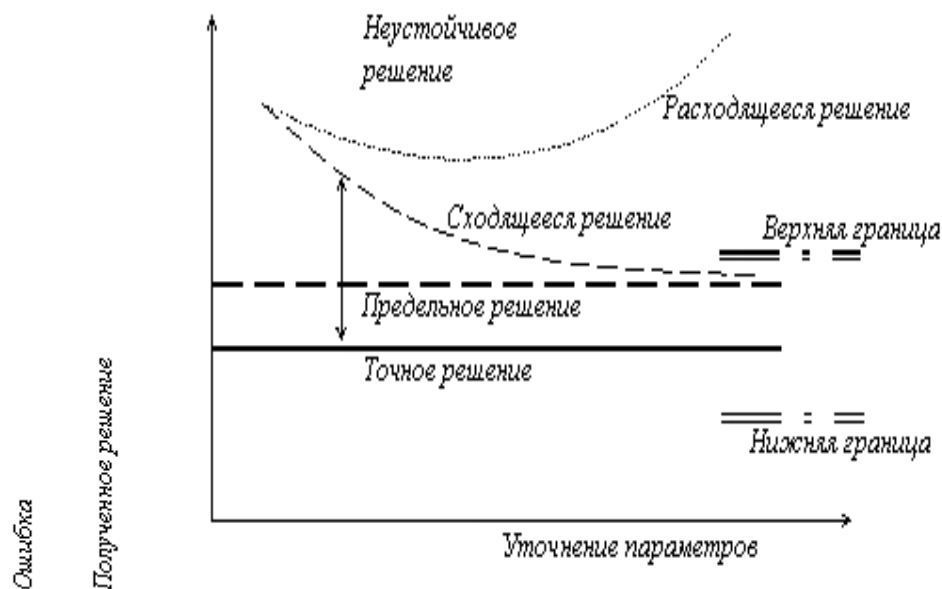


Figure 6 Solution dependency on parameters

Setting boundary conditions and material

By approximating the problem domain with a set of discrete finite elements, we must specify the characteristics of the material and the boundary conditions for each element. By specifying different characteristics for different elements, we can analyze the behavior of an object consisting of different materials.

According to the terminology of mathematical physics, which considers various differential equations describing physical fields, from a single mathematical point of view, the boundary or boundary conditions for these differential equations are divided into two main types: essential and natural. Usually, essential conditions are imposed on the desired function, and natural conditions on its derivatives with respect to spatial coordinates.

From the standpoint of the finite element method, essential boundary conditions are those that directly affect the degrees of freedom of the model and are superimposed on the components of the global vector of unknowns U (displacements). On the contrary, natural boundary conditions are those that indirectly affect the degrees of freedom through the global system of finite element equations and are superimposed on the right side of the system - the vector F (active forces).

In problems of mechanics, as a rule, those that include displacements (but not deformations, which are derivatives of displacements in spatial coordinates) are considered to be essential boundary conditions. According to the terminology of the theory of elasticity, such boundary conditions are called kinematic. For example, termination and articulation in rod problems are essential, or kinematic, boundary conditions imposed on the deflection or longitudinal displacement of the points of the rod. Note that in the problem of rod bending, the conditions imposed on the first derivative with respect to the longitudinal coordinate of the

rod deflection, which has the mechanical meaning of the angle of rotation of the rod section, also apply to essential conditions. The same can be said about the angles of rotation of the sections in the theory of bending plates.

The natural boundary conditions in mechanical applications of FEM include conditions imposed on various external force factors acting on points on the surface of the body - concentrated forces and moments in rod problems; distributed forces in two-dimensional and three-dimensional problems. Such restrictions are called force boundary conditions.

Mixed boundary conditions are widely used in the formulation of problems of continuum mechanics, and in particular the theory of elasticity. This means that at a given point on the surface of the body, several components of displacement and surface forces are simultaneously specified.

The listed three variants of boundary conditions are most common in purely mechanical applications of the FEM. In addition to the boundary conditions, to solve the equations, it is necessary to specify the characteristics of the material for each CE, from which the object of study is made. For example, in the study of the stress-strain state, parameters determine the relationship of stress and strain.

Formation of a system of equations

After specifying the boundary conditions and the material, the finite element analysis program forms a system of equations relating the boundary conditions with the unknowns, after which it solves this system with respect to the unknowns.

Getting the result

After finding the values of unknowns, the user can calculate the value of any parameter at any point of any finite element using the same unknown function that was used to build the system of equations. The output of the finite element analysis program is usually presented in numerical form. In problems of mechanics of solids, the output data are displacements and stresses. In heat transfer tasks, the output is temperature and heat flux through specific elements. However, using numerical data, it is difficult for the user to get a general idea of the behavior of the corresponding parameters. Graphic images are usually more informative, since they provide an opportunity to study the behavior of parameters throughout the entire task area.

Finite element method errors

The criteria of stability, convergence and accuracy are mainly determined by the errors of various kinds of operations carried out in FEM. Along with the usual rounding errors and the error of the approximate linear algebra methods used in FEM, there are also errors directly related to the finite element method. The division of the region into KE is not the only one. The dependence of the calculation on the user-made selection (construction) of the CE grid and the difficulty of assessing the accuracy of the results obtained are the main drawbacks of the method. The errors of the finite element method are related to:

- Discretization errors resulting from the differences between the actual geometry of the calculated area and its approximation by a system of finite elements;
- Approximation errors due to the difference between the actual distribution of the desired functions within the FE and their representation using the approximating functions.

3. CONCLUSION

The paper outlines some important factors of applying the FEM in MDTT, which must be taken into account when calculating a specific object.

REFERENCE

1. Norri D., Frieze T. Introduction to the finite element method. -M: Mir, 1986.-160c.
2. Segerlind L. Application of the finite element method. -M: Mir, 1979. -392s.
3. G. Streng, J. J. Fix. The Theory of the Finite Element Method. -M: Mir, 1977. -349s.
4. Bate K., Wilson E. Numerical analysis methods and finite element method. -M: stroiizdat, 1982.-448s.
5. Zenkevich O. Method of finite elements in engineering. - M: Mir, 1975. -541c.
6. Sabonnodier ZH.K. Pendant ZH.L. The finite element method and CAD. - M: Mir, 1979. -392s.
7. Rozin L.A. The finite element method as applied to elastic systems. - M: stroiizdat, 1977. -128c.
8. Postnov V.A., Kharkhurim I.Ya. The finite element method in the calculations of ship structures. -L.: Shipbuilding, 1974.-342c.